International Journal of Mechanical and Production
Engineering Research and Development (IJMPERD)
ISSN (P): 2249-6890; ISSN (E): 2249-8001
Vol. 8, Issue 2, Apr 2018, 307-314
© TJPRC Pvt. Ltd

MODELLING AND SIMULATION OF EFFECT OF COMPONENT STIFFNESS ON DYNAMIC BEHAVIOUR OF PRINTED CIRCUIT BOARD

P. K. DASH[1], M. MARY THRAZA[2] & BABITHAKODAVANLA[3]

[1] Professor, Department of Aeronautical Engineering, MLR Institute of Technology, Hyderabad, India

[2,3]Assistant Professor, Department of Aeronautical Engineering, Institute of Aeronautical Engineering Hyderabad, India

ABSTRACT

A spacecraft consists of a number of electronic packages to meet the desired functional requirements and in this, an electronic package is generally an assembly of printed circuit boards placed in a mechanical housing. A number of electronic components are mounted on the printed circuit board (PCB). A spacecraft experiences various types of loads during its launch such as vibration, acoustic and shock loads. Prediction of response for printed circuit boards due to vibration loads is important for mechanical design and reliability of electronic packages. The modelling and analysis of printed circuit boards are carried out for accurate prediction of response due to vibration loads using finite element method. A prediction about the vibration response of space bounded satellite's printed circuit board including the effect of component stiffness are coded using MatLab programming and the effect of contribution of component stiffness to the dynamic characteristics of PCB assembly is investigated. The analysis results are validated through vibration tests of PCB.

KEYWORDS: *Printed Circuit Board (PCB), Spacecraft, Vibration Analysis & Component Stiffness*

Received: Jan 19, 2018; **Accepted:** Feb 09, 2018; **Published:** Feb 28, 2018; **Paper Id.:** IJMPERDAPR201834

Original Article

INTRODUCTION

A spacecraft experiences various types of loads during its launch such as vibration, acoustic and shock loads. The electronic packages are designed to withstand the launch vibration environment. Electronic packages are subjected to vibration testing to establish adequate margins. Package component failures due to vibration loads have been observed in the past. The four basic failure modes of components mounted on PCB due to the random vibration environment are the results of the following conditions: high acceleration levels, high stress levels, large displacement amplitudes and electrical signals out of tolerance [1]. It is possible to predict the probability of mechanical failure by a two stage Physics of Failure (POF) approach. The first stage of this approach is defined as the response prediction stage. In this stage, vibration response of the board is calculated through a finite element (FE) model of the PCB component system. The second stage relates this calculated response to some pre-determined component failure criteria, to show whether the attached components can withstand this curvature or acceleration. Sophisticated electronic systems are often simulated using simple masses, springs and dampers to estimate the dynamic characteristics of the system. A simple one and two degrees of freedom systems are used to approximate the electronic systems. More complicated finite element models of electronic systems are created to study the dynamic characteristics of the system and to estimate the fatigue life of critical components mounted on the PCB. Finite element models can be either simplified or detailed. Detailed finite element models are built by modelling the PCB and the components. However, this approach is rarely used as it is time consuming and

expensive. Instead, simplified models of PCB are created where the components geometry is neglected. The component effects are included by increasing the Young's modulus and density of the PCB FE model, so it effectively behaves as if components were present. The simple geometry of the board is modelled and meshed using 2-D finite elements (i.e. by using flat shell elements). Sensitivity analysis of PCB finite element models was carried out by Amy et al. [2]. They determined the factors of safety by using different simplification methods of modelling the PCB. Pitarresi [3], Pitarresi, et al. [4], and Pitarresi and Primavera [5] provided the solutions for issues encountered in modelling the PCB assembly that includes a wide variety of components. In this paper, modelling and simulation of a typical component mounted on PCB used for space applications is carried out. First, vibration analysis of a bare PCB is carried out using FEM to determine the natural frequencies. The PCB is modelled using shell elements. The FEM model is validated by conducting vibration tests on the PCB and comparing the simulation and test results. Next, static analysis of the component mounted on PCB is carried out to determine the contribution of component stiffness to the PCB. The effect of the component stiffness to the PCB is calculated in terms of stiffness coefficients of the PCB based on this analysis. The stiffness coefficients give the effective stiffness of the PCB that includes the effect of component stiffness. The component is modelled using beam and shell elements. Subsequently, modal analysis and frequency response analysis are carried out for a PCB with components by using the stiffness coefficients derived from the static analysis.

EXPERIMENTAL SETUP AND METHOD

In this study, a six-layer PCB used for space applications is considered. The PCB is modelled as isotropic plate with equivalent material properties such as Young’s modulus, Poisson’s ratio and mass density. Details of the PCB are summarized in Table 1. The PCB is modelled using PATRAN as pre-processor and MSC. NASTRAN is used as solver. The PCB is meshed with 1800 quadrilateral shell elements with appropriate thickness. Fixed /clamped boundary conditions are applied at nine locations (PCB mounting locations) by arresting six degrees of freedom for the nodes on the boundary of holes in PCB as shown in Figure [1].

Table 1: Details of PCB

Parameter	Value
PCB size	250×200×2.1 mm
Mass of Bare PCB	208.4 gm
Young’s modulus of Bare PCB	20 GPa
Poisson’s ratio	0.12
Boundary Condition	Fixed/clamped

FE Simulation Model for Bare PCB

Normal mode analyses were conducted on FE model to extract first three fundamental natural frequencies for bare PCB. The calculated first three natural frequencies are 318.7 Hz, 354.1 Hz and 368.1 Hz for bare PCB. Mode shapes corresponding to these frequencies are given in Figures [2] to [4].

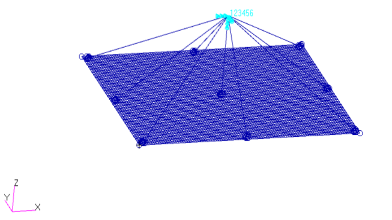

Figure 1: FE Model of Bare PCB

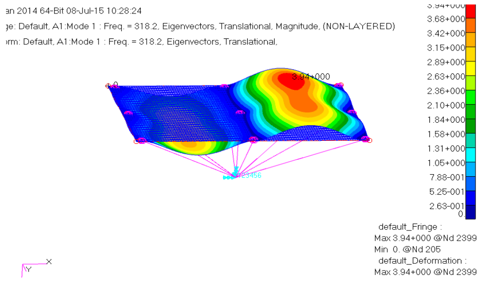

Figure 2: Mode Shape of Bare PCB for First Frequency

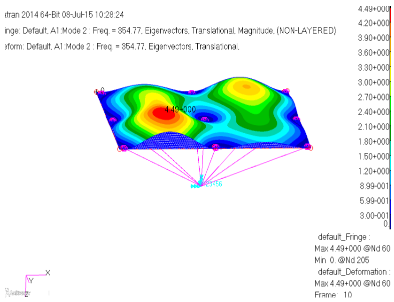

Figure 3: Mode Shape of Bare PCB for Second Frequency

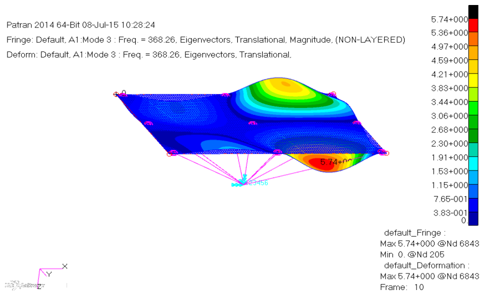

Figure 4: Mode Shape of Bare PCB for Third Frequency

Experimental Test Setup for Bare PCB

The vibration test was conducted by mounting the PCB with screws at nine locations on a vibration table. The bare PCB mounted on the vibration table is shown in Figure [5]. Accelerometers are mounted at various locations of the PCB to measure the responses. The vibration test was carried out in the vibration test facility consisting of electro-dynamic shaker, control system, signal conditioners and data acquisition system. The frequency response function (FRF) is obtained using an electro-dynamic shaker by conducting a sine sweep test. In sine sweep test, the input acceleration is given to the test specimen using electro-dynamic shaker and the output acceleration at various desired locations of the test specimen is measured using accelerometer. The ratio of output to input acceleration gives the FRF at that location. The experimental frequency response plot for bare PCB at a specific location is shown in Figure [6].

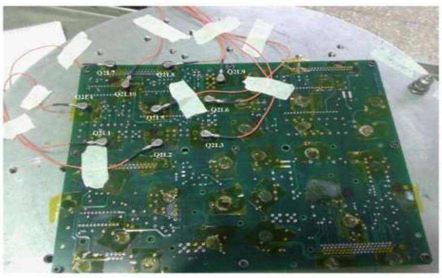

Figure 5: Bare PCB on Vibration Table

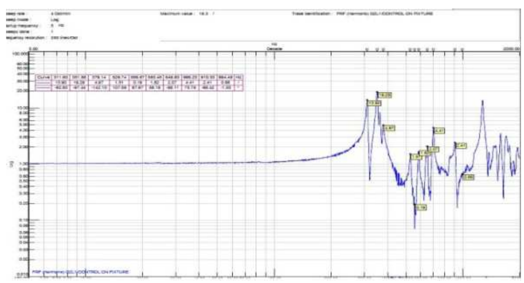

Figure 6: Experimental Frequency Response Plot for Bare PCB

FEM Model Validation

The FEM model is validated by comparing the FEM simulation results and the experimental test results. Simulation and test results for fundamental frequencies of the bare PCB are compared in Table 2. The simulation and test results for bare PCB are matching well. Hence the FEM model is validated.

Table 2: Comparison of Fundamental Frequencies for Bare PCB

Frequency (Hz)	Simulation Results	Test Results	% Difference
1	318.7	311	3.76
2	354.1	351	2.53
3	368.1	379	0.5

Static Analysis of a PCB with Component

Static analysis of the component mounted on PCB is carried out to determine the contribution of component stiffness to the PCB. The effect of the component stiffness to the PCB is calculated in terms of stiffness coefficients of the PCB based on this analysis. A typical component used in space applications is considered. The component consists of casing and the terminals (pins) as shown in Figure [7]. The component is mounted on the PCB by inserting the terminals in plated through holes and then soldering the terminals on PCB. The physical and material properties of the component are given in Table 3 and Table 4. The component casing is modelled using shell elements and terminals using beam elements.

The static analysis is carried out for a standard PCB size used for 3-point bending test. In 3-point bending test, the PCB is simply supported at the ends and the load is transversely applied in the middle of the PCB. First, the deformation is determined at the mid-point for a bare PCB and next for PCB with the component. Figure [8] shows the deformation plot of PCB with the component. The ratio of the deformations for first to second case gives the stiffness coefficient. The stiffness coefficient dues the effective stiffness of PCB that includes the effect of component stiffness the effective stiffness of PCBs can be used in the local smearing approach at the component foot print location.

Table 3: Details of Component and Terminals

(L*B*H) Of Component	20.57*10.41*2.1
Length of the terminal	3mm
Diameter Of Terminal	0.7mm

Table 4: Properties of Casing and Terminals

Location	Casing	Terminal
Young’s modulus	70GPa	159GPa
Density	8070kg/m^3	8000kg/m^3
Poisson’s ratio	0.33	0.33

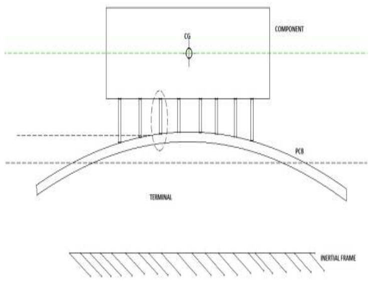

Figure 7: Deformation of PCB with Component

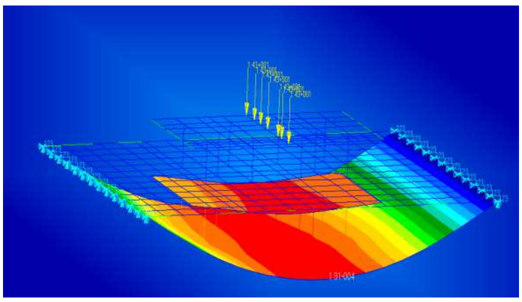

Figure 8: Deformation Plot of PCB with Component

Table 5: Linear Static Analysis of PCB With and Without Component

Analytical Deformation	Bare PCB (Without Component)	With Component	Stiffness Coefficient
2.54-004	2.65-004	1.31-004	2.02

Vibration Analysis of a PCB with Component

In this section, modal analysis and frequency response analysis are carried out for a PCB with component. The component considered for static analysis is also taken for vibration analysis. The analysis is carried out for two cases. In the first case, the detailed modelling of the component with PCB is carried out. In the second case, the stiffness and mass of the component are simulated locally on the PCB.

Detailed Modelling of PCB with Component

In this section, detailed modelling of the component is carried out. The component casing is modelled using shell elements and terminals using beam elements. The FE model of PCB with component is shown in figure [9]**.** Normal mode analysis was conducted on FE model to extract first three fundamental natural frequencies for PCB with component. The calculated first three natural frequencies are 324.0 Hz, 360.1 Hz and 385.0 Hz for PCB with component. Mode shapes corresponding to these frequencies are given in figure [10 -12].

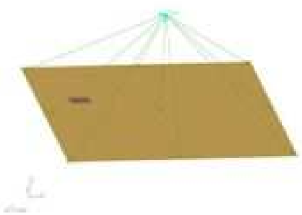

Figure 9: FEM Model of PCB with Component

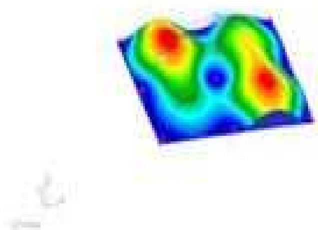

Figure 10: Mode Shape of PCB with Component for First Frequency

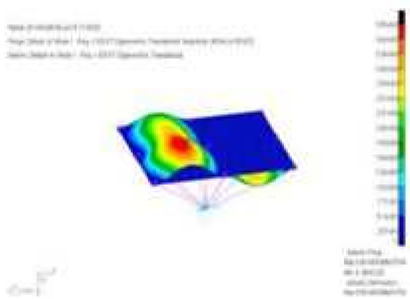

Figure 11: Mode Shape of PCB with Component for Second Frequency

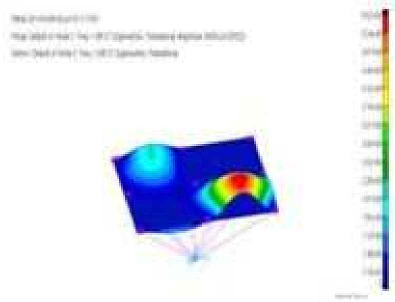

Figure 12: Mode Shape of PCB with Third Frequency

Modelling of PCB with Component using Local Smearing Approach

In this section, the stiffness and mass of the component is simulated on the PCB using local smearing approach. An example of a locally smeared FE model of a PCB is shown in Figure 13. The effect of component on PCB is modelled

by increasing the stiffness of PCB over the footprint of the component. The PCB density over the component footprint includes the density of the component. The equivalent Young's Modulus of the PCB at the component footprint is given by Young's Modulus of the bare PCB times the stiffness coefficient derived from the static analysis. Hence, Young's Modulus of the PCB at the component footprint = 20*2.02=40.4 GPa.

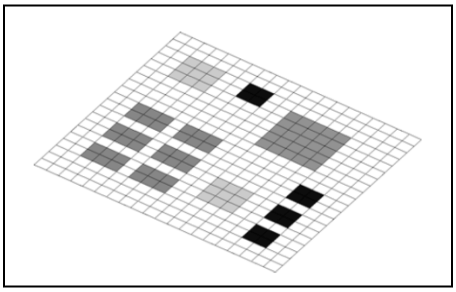

Figure 13: Example of a Locally Smeared FE Model of a PCB

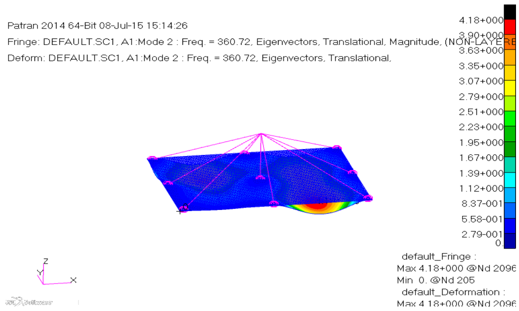

Figure14: Mode Shape of Locally Smeared PCB First Frequency

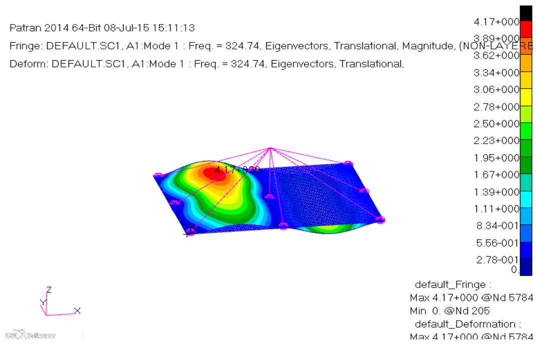

Figure15: Mode Shape of Locally Smeared PCB Second Frequency

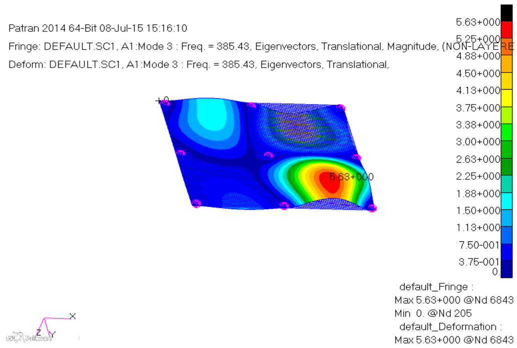

Figure16: Mode Shape of Locally Smeared PCB Third Frequency

RESULTS AND DISCUSSIONS

In this section, the results of the detailed modelling approach and the local smearing approach are compared. The natural frequencies of the PCB for two approaches are compared in Table 6. The results are matching well. The maximum FRF for the second natural frequency at the component location is shown in Figure 17 and Figure 18 and compared in Table 7. These results also show good agreement. Hence the local smearing approach is also appropriate for determination of natural frequencies and the FRF. This is especially useful for modelling of PCB mounted with a number of components. Detailed component modelling, which is time consuming can be avoided. Instead, local smearing approach can be applied based on the stiffness coefficients obtained for the components. The stiffness coefficients can be obtained by simulation or experiments for different type of components.

Table 6: Comparison of Natural Frequencies of PCB with Component for different Approaches

Mode	Natural Frequency (Hz) Detailed Modelling	Natural Frequency (Hz) Local Smearing
1	328	324
2	363	360
3	385	385

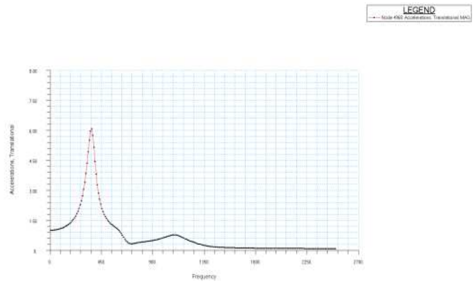

Figure17: FRF Plot at Component Location for Detailed Modelling Approach

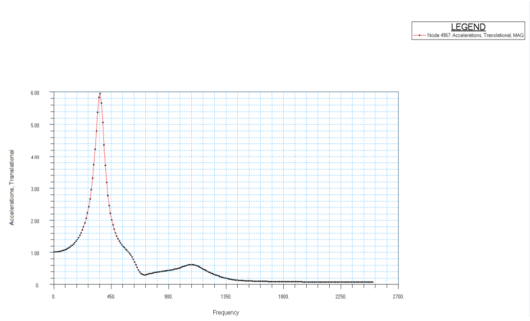

Figure18: FRF Plot at Component Location for Local Smearing Approach

Approach	FRF
Detailed Modelling	6.0
Locally Smearing approach	5.9

Detailed Modelling for Component Stresses/Strains

For determination of stresses/strains for the component or at the PCB-component interface, detailed modelling approach is required. Stresses/strains for a base excitation of 100 $m/s^2$is determined using detailed modelling of the component on the PCB. The strain plots for the PCB and maximum strains at the PCB - terminal interface are shown in Figures 19-20. The maximum strains at the PCB - terminal interface occur from the outer terminals of the component. The maximum strains at the PCB - terminal interface for first three natural frequencies are shown in Table 7. The strains are maximum for the third natural frequency. The stress plots for the PCB and maximum stresses at PCB-terminal interface are shown in Figures 21-22. The maximum stresses (axial and bending) for component terminal for first 3 natural frequencies are shown in Table 8. The terminal's maximum bending stress occurs in the outer terminal for the third natural frequency.

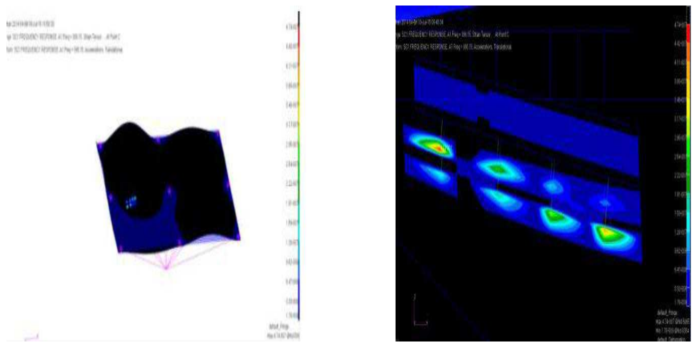

Figure 19: Stain Plot for PCB and Maximum Strain at PCB-Terminal Interface

Table 8: Comparison of Maximum Strains at PCB-Terminal Interface for Natural Frequencies

Mode No	Natural Frequency (Hz)	Maximum Strains at PCB - terminal interface (μ ε)
1	326	18.6
2	361	32.6
3	386	47.4

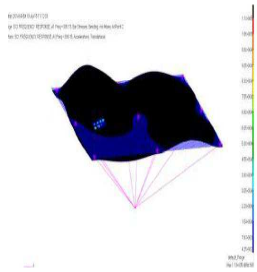 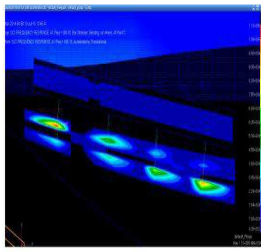

Figure 20: Stress Plot for PCB (left) and Maximum Stress at PCB Terminal Interface (Right)

Table 9: Comparison of Maximum Stresses for Component Terminal for Natural Frequencies

Mode No	Natural Frequency (Hz)	Terminal Maximum Axial Stress (MPa)	Terminal Maximum Bending stress (MPa)
1	326	1.95	4.44
2	361	2.57	7.77
3	386	1.93	11.3

CONCLUSIONS

Vibration analysis of a printed circuit board is carried out including to the effect of component stiffness. For model validation, the FEM simulation results are compared with experimental test results for bare PCB. The vibration analysis of a typical PCB mounted with a component is carried out using two different approaches: detailed component modelling approach and local smearing approach. For local smearing approach, the effect of the component stiffness to the PCB is calculated in terms of stiffness coefficients of the PCB based on static analysis. The results of the detailed modelling approach and the local smearing approach are matching well for the natural frequencies and FRF. Hence, local smearing approach is appropriate for determination of natural frequencies and the FRF, since detailed component modelling approach is time consuming. Detailed component modelling approach is required for determination of stresses/strains for the component or at the PCB-component interface. The maximum stresses and strains at PCB-terminal interface occur for the outer terminals of the component.

REFERENCES

1. ***Steinberg D. S***., *Vibration Analysis for Electronic Equipment. 3rd ed. New York: John Wiley & Sons Inc., 2000.*
2. ***Amy R. A., Aglietti G. S., Richardson G***., *"Sensitivity analysis of simplified Printed Circuit Board finite element models", Microelectronics Reliability, Vol. 49, pp. 791-799, 2009.*
3. ***Pitarresi, J.M***., *"Modelling of Printed Circuit Cards Subject to Vibration", IEEE Proceedings of the Circuits and Systems Conference, New Orleans, LA, May 3-5, pp. 2104-2107, 1990.*
4. ***Pitarresi J.M., Celetka D., Coldwel R. and Smith D***., *"The Smeared Properties Approach to FE Vibration Modelling of Printed Circuit Cards", ASME Journal of Electronics Packaging, Vol.113, pp. 250-257, 1991.*
5. ***Pitarresi J.M. and Primavera A***., *"Comparison of Vibration Modelling Techniques for Printed Circuit Cards", ASME Journal of Electronics Packaging, Vol.113, pp. 378-383, 1992*